Optimization of heat transfer using CFD simulation for concentric helical coil heat exchanger for constant temperature outer wall

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

BY

SAGAR DAS

110ME0287

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA – 769008

(2013-2014)

Optimization of heat transfer using CFD simulation for concentric helical coil heat exchanger for constant temperature outer wall

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

BY

SAGAR DAS

110ME0287

Under the guidance of

Prof. ASHOK K. SATAPATHY

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA – 769008 (2013-2014)

CERTIFICATE

This is to certify that the thesis entitled **"Optimization of heat transfer using CFD simulation for concentric helical coil heat exchanger for constant temperature outer wall"** submitted by **Sagar Das (Roll no. 110ME0287)** in partial fulfillment of the requirements for the award of Bachelor of Technology degree in Mechanical Engineering at the National Institute of Technology, Rourkela is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University/Institute for the award of any Degree or Diploma.

Prof. Ashok K. Satapathy
Dept. of Mechanical Engineering
National Institute of Technology
Rourkela – 769008

ACKNOWLEDGEMENT

I wish to express my profound gratitude and indebtedness to **Prof. Ashok K. Satapathy**, Department of Mechanical Engineering, NIT, Rourkela for introducing the present topic and for his inspiring guidance, endless support and valuable suggestion throughout the project work. I am extremely fortunate to be involved in an exciting and challenging research project like "**Optimization of heat transfer using CFD simulation for concentric helical coil heat exchanger for constant temperature outer wall**". It has enriched my life, giving me an opportunity to work in a new environment of Fluent. This project increased my thinking and understanding capability as I started the project from scratch. He has not only been a wonderful supervisor but also a genuine person. I consider myself extremely lucky to be able to work under the guidance of such a dynamic personality. Actually he is one of such genuine person for whom my words will not be enough to express.

I am also thankful to **Dr. K. P. Maity, H.O.D** of Department of Mechanical Engineering, National Institute of Technology, Rourkela for his constant support and encouragement.

Last, but not the least I extend my sincere thanks to all faculty members of Mechanical Engineering department for making my project a successful one, for their valuable advice in every stage and also giving me absolute working environment where I unlashed my potential. I would like to thank all whose direct and indirect support helped me completing my thesis in time. I want to convey my heartiest gratitude to my parents for their unfathomable encouragement.

Sagar Das

110ME0287

Bachelor of Technology,

Mechanical Engineering Dept.

CONTENTS

List of Tables:

ABSTRACT

Thermodynamic Optimization in heat transfer of a concentric coiled tube-in-tube heat exchanger under constant wall temperature condition, based on Fluid–Fluid heat transfer is focused in this paper. The parameters which influence the nature of flow in a helical coil are pitch coil diameter, pitch and tube diameter in helical coils. CFD analysis was carried out and their variation on thermal and hydraulic characteristics were analyzed, with varying Reynolds number (hot fluid) and varying tube-to-coil diameter ratios for a given flow velocity of cold fluid. The analysis was carried with Ansys 13.0 Fluent, for turbulent counter-flow with fluid water. The correlations for heat transfer and drop in pressure were analyzed. Thus, Nusselt number and friction factor were also calculated. Graphs were plotted between Nusselt number, friction factor, pressure drop and power loss with Reynolds number. The point where the friction fraction intersects with the Nusselt number is the point where the heat transfer is optimum, corresponding to that Reynolds number. Beyond that Reynolds number, the friction factor decreases rapidly, hence the pressure drop increases and so the power loss also increases. Various velocity and temperature contours were also obtained. Hence, we found the optimum value of Reynolds number for the corresponding tube-to-coil diameter ratios. It thus minimizes the degradation of thermal energy and viscous dissipation of mechanical energy.

NOMENCLATURE

- f = Darcy's friction factor
- $m^{\cdot}$ = mass flow rate
- A = area of heat transfer (m^2), $A = (\pi * d^2)/4$, m^2
- D = helical Coil diameter
- h = heat transfer coefficient ($W\ m^{-2}\ K^{-1}$)
- H = helical pitch (m)
- K' = thermal conductivity ($W\ m^{-1}\ K^{-1}$)
- L= length of the pipe (m)
- Nu = Nusselt number
- Pr = Prandtl number
- q = heat transferred (W)
- $R\ i$ = radius of the inner tube (m)
- R_{out} = radius of the outer tube (m)
- R = resistance the flow of thermal energy ($W^{-2}\ m^2\ K$)
- Rc = pitch circle radius of the pipe (m)
- Re= Reynolds number
- v = velocity (ms^{-1})
- U = overall heat transfer coefficient ($Wm^{-2}\ K^{-1}$)
- V = volume (m^3)
- O = helix angle (rad)
- δ = curvature ratio = D/d
- Δ = temperature difference (K)
- μ = viscosity ($kg\ m^{-1}\ s^{-1}$)
- ρ = density ($kg\ m^{-3}$)
- LM = logarithmic mean
- w = power loss

CHAPTER 1

INTRODUCTION

1.1. Heat Exchangers

Heat transfer between the flowing fluids is an important physical process to be focused upon, thus heat exchangers plays a vital role in our day to day life. They have a wide range of application like they can be used in a varied type of installations, food processing, transportation, domestic applications, process industries, nuclear power plant, air conditioning, compact heat exchangers, recovery process, HVACs, refrigeration, etc.

The main purpose of the heat exchanger is to frame an efficient method of heat transfer/exchange from one fluid to another, by simple direct or indirect contact. The heat transfer mostly occurs by three principles i.e. convection, conduction and radiation. The heat transfer through radiation in a heat exchanger is generally not taken into account, as it is comparatively negligible to the heat exchange by conduction and convection. The process conduction occurs when there is a temperature gradient between the solid wall. It can be maximized by selecting a critical radius of insulation of the wall and a high conductive material. Convection plays an important role in the heat exchanger performance. Forced convection accelerates the heat transfer in a heat exchanger from a moving stream of fluid to the wall of pipe or vise-versa.

In the applications of heat exchanger, improvement is focused on the efficiency, substantial cost, material saving and space.

1.2. Types of Heat Exchangers [8]

1. Recuperators or Transfer type heat exchanger
2. Regenerators or Storage type heat exchanger
3. Mixers or Direct contact type heat exchanger

1.3. Flow arrangements in Recuperative heat exchangers [8]

1. Parallel flow heat exchanger
2. Counter flow heat exchanger
3. Cross flow heat exchanger

1.4. Tubular Heat exchanger

They are mainly of circular cross section. The circular cross section provides flexibility in the design parameters like length, diameter, thickness of tube and their arrangement can be modified easily and can be arranged to multiple complex shapes. It is mostly used for single phase i.e. liquid-to-liquid heat transfer. It is further classified into:

1. Shell and tube heat exchanger
2. Double pipe heat exchanger
3. Spiral tube heat exchanger

1.5. Helical tube-in-tube Heat exchanger:

The tube-in-tube helical heat exchanger consists of one tube oriented concentrically inside another with a greater radius. The flow configuration may be of the following two types i.e. parallel or counter flow. It can also be arranged in a lot of parallel and series configurations to meet different requirements of heat transfer. The helical arrangement stands out to be used in various industrial applications. Though this configuration has been widely used, through knowledge should be focused upon the heat transfer coefficient, temperature gradient, pressure drop with various flow patterns are of much importance. The curvature forms a secondary flow in the tubes, which is just normal to the direction of flow to the primary axis. Heat transfer occurring between the wall and the fluid in increased substantially by this secondary flow, which also offers a greater area for the heat transfer within a small compact space, with higher heat transfer coefficient. Types of flow in curved pipes has been focused and effect of Prandtl number and Reynolds number has been related on the following flow patterns and also on Nusselt number.

1.6. Characteristics of Helical Coil [1]

The helical coil has a pitch of height 'H', diameter of tube as '2r', the coil diameter '2Rc', curvature ratio as 'i' i.e. ratio of tube and coil diameter as 'r/Rc'. The helix angle '2a' is the angle between its projection on a surface and measuring angle between the coil, shown by 'v' shape.

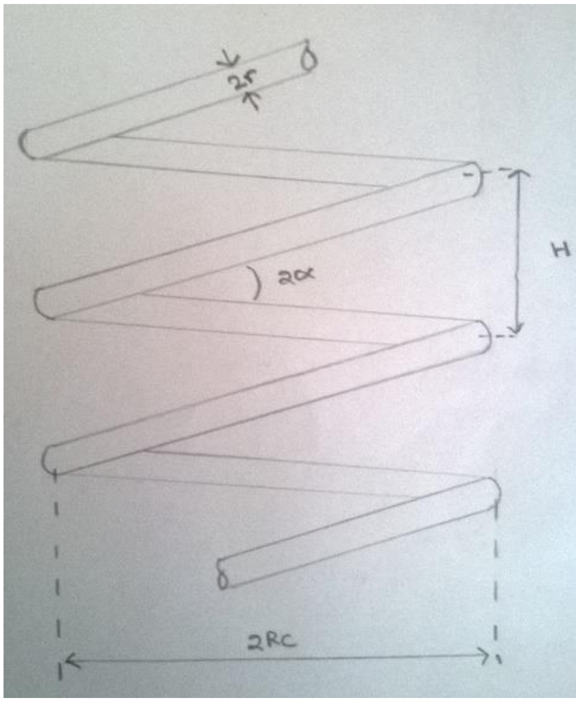


Fig. Geometry of a helical coil

The centrifugal force is governed by the curvature of the tubular coil, while torsion occurs due to the helix angle or pitch. The centrifugal force leads to the development of secondary flow in the helical heat exchanger.

1.7. Advantages & Disadvantages:

Advantages of helical coil:

1. Helical tubes have high heat transfer co-efficient compared to straight tubes and are more compact.
2. It increases the contact area and time for heat exchange between the two fluids, which leads to higher rate of heat exchange.
3. The tube curvature causing secondary flow pattern, which is perpendicular to the main stream flow acts as an additional convective heat transfer mechanism.
4. Coils generally give better heat transfer performance, as they do have higher process side coefficient and lower wall resistance.
5. The entire surface area of the curved helical tube is exposed to moving fluid, which thus eliminates any dead-zones, which is a common drawback in the shell-tube type heat exchanger.
6. The spring-like coil in helical heat exchanger eliminates any thermal expansion and thermal shock problems, which has its application in high pressure operations.
7. Fouling is comparatively less in helical coil type than shell and tube type because of greater turbulence created inside the curved pipes.

Disadvantages of coils:

1. Design of helical tube-in-tube heat exchanger is complex.
2. Cleaning of the tubes are more difficult than jackets and shells.
3. For high reactive or corrosive fluids, coils can't be used, instead jackets are preferred.
4. Coils generally plays a major role in the selection of any agitation system, as densely packed coils may create some unmixed regions by any interference with the fluid flow.

1.8. CRITICAL RADIUS OF INSULATION OF CYLINDRICAL SURFACE [9]

Heat transfer decreases when we add insulation to the tubular pipe. Heat transfer rate decreases to a greater extent, as we increase the thickness of insulation. If the surface or area of heat transfer is kept constant and insulation is added, then it gradually increases the thermal resistance, thus increases thermal resistance.

The resistance of conduction increases as the addition of the insulation layer, but at the same time the convection resistance decreases because of the notifying increase in the outer surface area for heat transfer due to convection. The heat transfer from the pipe may decrease or increase, depending on which effect, i.e. convection or conduction dominates. The rate of heat transfer to the surrounding atmosphere from the insulated pipe is expressed as:

$$Q = (T_1 - T_0) / (R_{ins} - R_{conv})$$

$$= (T_1 - T_0) / [(\ln (R_2/R_1) / 2 * 3.14 * L\, k) + 1/(h * 3.14 * R_2 L)]$$

We know, that the critical radius (R_{cr}) of insulation depends on:

1. The thermal conductivity of insulation ‘k’ and
2. Convection heat transfer coefficient ‘h’.

Rate of heat transfer increases from the cylindrical surface with the increase of insulation for $R2 < R\text{cr}$, which reaches a maximum value when $r2 = r\text{cr}$, and then starts to decrease from $r2 > r\text{cr}$. It is thus clear that the value of critical radius Rcr , will be large when k is high and h is less.

So, for a cylindrical surface the following relation holds:

$$R_{cr,\ max} = k_{max,\ insulation} / h_{min}$$

1.9. <u>Aim of the Present Work:</u>

Numerical analysis were carried out to determine heat transfer characteristics for the tube-in-tube helical heat exchanger. CFD analysis was carried out and their variation on thermal and hydraulic characteristics were analyzed, with varying Reynolds number (hot fluid) and varying tube-to-coil diameter ratios for a given flow velocity of cold fluid. The analysis was carried with Ansys 13.0 Fluent, for turbulent counter-flow with fluid water. The correlations for heat transfer and drop in pressure were analyzed. The objective of the project is also to obtain a better and quantitative insight to the heat transfer process that generally occurs when a fluid flows inside a helical coiled tube. The materials were decided and fluid taken was water and the material for the pipe was taken as copper because it has better conducting properties.

To optimize the heat transfer coefficient by plotting graphs between:

a) Friction factor and Reynolds number
b) Nusselt number and Reynolds number
c) Decrease in pressure and Reynolds number
d) Power loss and Reynolds number
e) Temperature profile and velocity profile

1.10. <u>**Methodology**</u>:

- Literature Review
- Project definition and description
- Selection of variation of parameters of tube.
- Selection of material along with specifications such as thickness and size.
- Boundary conditions: Inlet (velocity, mass flow rate, hydraulic diameter for backflow parameters), Outlet (Pressure), wall (constant temperature and no slip condition)
- Observing and analyzing the characteristics of flow.

.

CHAPTER 2

LITERATURE REVIEW

S.D. Sancheti & DR.P.R .Suresh have worked on the Experimental and CFD estimation of the heat transfer in helically coiled heat exchanger. His work focused on the fluid – to – fluid heat transfer. He validated the basic methodology of CFD analysis in a heat exchanger, without considering actual properties of fluid, a constant value was established instead. For various boundary conditions, the heat transfer characteristics was compared for a helical coil. He found that specification for constant heat flux and constant temperature boundary condition doesn't yield desired modeling for an actual possible heat exchanger. So, heat exchanger was analyzed considering conjugate temperature dependent and heat transfer properties. The fabrication of an experimental set up for the heat transfer characteristics was developed. Experimental results were compared with the results of CFD calculation using CFD package i.e. FLUENT 6.2. Use of constant values for the basic thermal and transport properties in the heat exchanger resulted in prediction of inaccurate heat transfer coefficients. From the results obtained from experiment a correlation was developed for the calculation of inner heat transfer coefficient in a helical coil heat exchanger. [1]

Rahul Kharat, Nitin Bhardwaj and R. S Jha worked on the Development of heat transfer coefficient correlation for concentric helical coil heat exchanger. The existing correlation was found to result in a large discrepancies with increase in the gap between the two concentric coils when they were compared with the experimental results. In their work, CFD simulations and with Fluent 6.3.26 was compared with the experimental data, which was used to develop improved correlation for the heat transfer coefficient. Mathematical model was also developed for analyzing the data, which were obtained from experimental results and CFD to accounts for the effects, by different parameters and functional variables like coil gap, coil diameter and tube diameter. Using numerical technique Optimization was done for the heat transfer coefficient by the new correlation, which fits the experimental data within an error band of 3-4%. [2]

J.S. Jayakumar , S.M Mahajani, J.C Mandal, Kannan N. Iyer and P.K. Vijayan worked on the Thermal hydraulic characteristics of air-water two-phase flows in helical pipes. He worked on the Two fluid Eulerian-Eulerian calculation in Fluent 6.3 for the analysis. The parameters which influence the nature of flow are pitch coil diameter, pitch and diameter in helical coils. CFD analysis was carried out and their variation on thermal and hydraulic characteristics by changing the inlet void fraction for a given flow velocity. The correlations for heat transfer and drop in pressure were analyzed. Estimation of inner heat transfer coefficient by changing the void fraction and flow velocity, results in reduction in ‘h’ is less below 5% and significant above 15% void fraction [3]

N. Ghorbani, H. Teherain, M. Gorhi and H. Mirgolbabaei worked on the Experimental study of mixed convection of heat exchanger in vertically concentric helical coil heat exchanger. Mixed convection heat transfer in a coil in cell heat exchanger with varying Reynolds no. with varying tube-to-coil diameter ratios and coil pitch were investigated experimentally, for both laminar and turbulent flow. The effects of the coil pitch and diameter of tube on shell-side heat transfer coefficient were studied of the helically coil heat exchanger. Nusselt number correlation with variable coil parameters were analyzed to best fit the data. It is compared with similar studies with specific boundary conditions. It, concluded that, the tube diameter has negligible influence on heat transfer on shell- side, coil surface has –ve effect on h_0. The overall heat transfer increases with h_0. [4]

Timothy J. Rennie, Vijaya G.S. Raghavan worked on the Numerical studies of a double pipe helical heat exchanger. A double-pipe helical heat exchanger was modeled numerically for numerical flow and the heat transfer characteristics were studied for different fluid flow rates and tube sizes. The overall heat transfer coefficient for both counter and parallel flow were calculated. Simulations were validated by comparing the Nusselt numbers. Greatest thermal resistance were found around the annular region. A correlation was found between the annulus Nusselt no. with a modified dean noumber, which gave a strong linear relationship. [5]

Akiyama, M. and Cheng, K.C. worked on the Boundary vorticity method for laminar forced convection heat transfer in curved pipes. Based on their theoretical work he proposed that the Vortex formation and conversion from laminar to turbulent flow occurs due to the secondary flow by the curvature of the helically coil, which produces a centrifugal force and was analyzed by the CFD using Fluent and various correlations were developed. [6]

Dr. Ashok K. Satapathy worked on the Thermodynamic Optimization of a coiled tube heat exchanger under constant wall heat flux condition, based on Fluid – Fluid heat transfer. Optimum diameter ratio of coil, minimizes the degradation of thermal energy and viscous dissipation of mechanical energy. [7]

CHAPTER – 3

CFD MODELING

Computational fluid dynamics (CFD) is the study of system, which starts with the basic construction of the desired geometry and then meshing for modeling the desired dominion. Generally, the geometry is thus simplified for CFD studies. Mesh is the discretization/division of the domain into further small volumes where then the equations are simplified and solved by the help of simple iterative methods. Modeling starts with describing the boundary and the initial conditions for the domain and leads to the modeling of the total system. It is thus followed by analysis of the obtained results, discussions, calculations and conclusions.

3.1. Geometry:

Heat exchanger with a concentric tube-in-tube is created in the ANSYS 13.0 workbench design module. It is a heat exchanger, with counter-flow. Initially, the fluid flow (fluent) module is selected from the workbench. The design model opens up as a new window on clicking the geometry option. Scale is taken in mm.

3.1.1. Sketching

Initially, there are 3 reference planes, i.e. XY-plane, YZ-plane and ZX-plane. Any of the plane is selected say the XY-plane is selected. Then the sketch button which appears on the toolbar menu is selected 5 times as 5-sketches are to be drawn and then integrated to form the geometry, which comprises of 4 circles and a line with required boundary conditions to constraint the sketch.

So, the first sketch is selected from the tree outline and then selected the sketching button and a circle is selected with the center and radii needed to constraint it. The circle is drawn on the +ve, X-axis and then selected the modeling option to constraint it by giving the radii as 5 mm and distance from the center of circle to the origin as 50 mm (taken D/d = 10) i.e. ratio of coil diameter and tube diameter as 10 in the first case.

This process is followed by the next 3 concentric circles with radii as 6 mm, 11 mm and 12 mm. The 5-6 mm is the thickness of the inner tube and 11-12 mm is the thickness of outer tube. The 5^{th} sketch drawn is a line on the +ve Y-axis starting from the origin with the height = (length of pitch of helix * no. of turns). So, here 1.5 turns and pitch = 30 mm is taken, so height = 45 mm is taken. Now, generate button is clicked. So, the basic sketch is created.

3.1.2. Sweeping

The Sketch 1, 2, 3 & 4 are swept along the path, i.e. line drawn in sketch 5, by editing the details view on the tray by changing the operation to “add frozen” to construct the 3D model. The helical sweep is of 1.5 turns as the twist specification which is defined as number of turns. Then the helical tubes are created by generating the sweep.

3.1.3. Boolean Operation

The Boolean operation is selected from the create toolbar 3 times as four volumes are to be created without any volume being intersected. So, the 1^{st} Boolean operation is selected and subtract operation is selected with 4^{th} volume – 3^{rd} volume, with 3^{rd} volume retained. Likewise the other Boolean operations are performed.

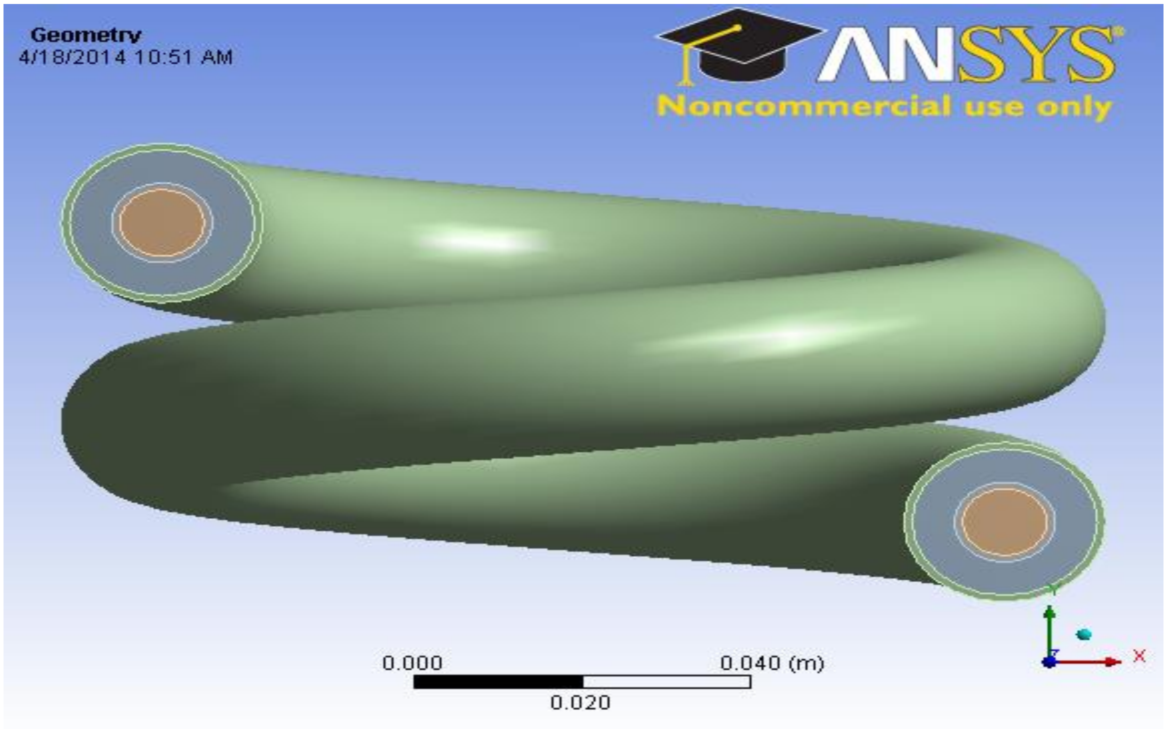
3.1.4. Merging

After sweep and Boolean operation, it shows that the model has 4 parts and 4 bodies. So, in the merge operation, all the 4 parts are then selected using control and selecting new parts and thus merged as 1 part. Then, it will show1 part and 4 bodies. The 4 bodies obtained within 1 part are thus named and edited the type fluid/solid as follows:

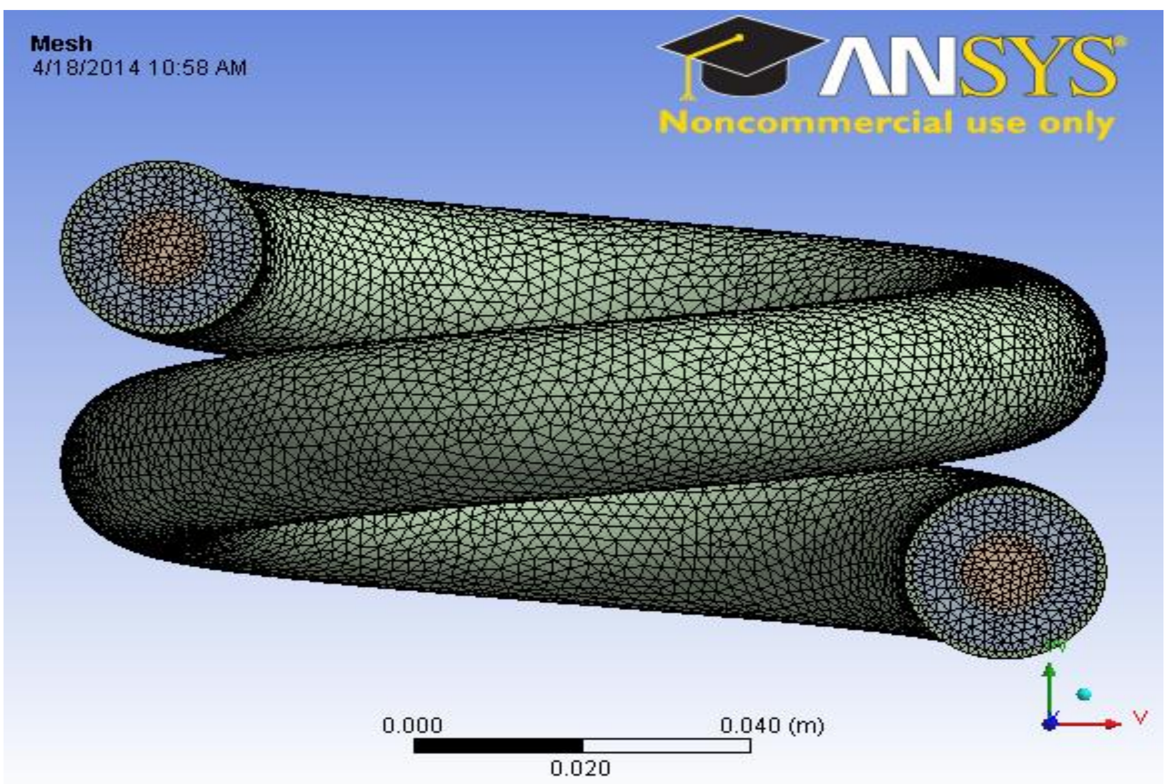
Table 1. Names of parts of the body

Naming of various parts of the body with state type Part number	Part Of The Model	State Type
1	Inner_Fluid	Fluid
2	Inner_Pipe	Solid
3	Outer_Fluid	Fluid
4	Outer_Pipe	Solid

Thus, the geometry is created which is then generated, refreshed and then updated.
Save the project.



Geometry of Helical coil with 1.5 turns



Meshing of helical coil

3.2. Meshing :

This mesh generally contains mixed cells (i.e. Tetra and Hexahedral cells) with both triangular faces and quadrilateral faces at boundaries. Care should be taken to use the structured hexahedral cells to an extent as high as possible. A fine mesh is thus generated. For this fine meshing, the edges and the regions of preferably high temperatures and pressure gradient are meshed finely.

3.2.1. Mapped Face Meshing

The mesh can be applied by creating or clicking on mapped face meshing 4 times, to create 4 mapped face mesh. The opposite faces of the inlet and outlet are selected and applied after one another.

3.2.2. Sizing

Then the edge sizing is defined for the no. of divisions we want to divide the entire volume into small parts to perform the finite element analysis. Each edge is defined with no. of even divisions increasing with the edge diameter.

3.2.3. Named Selection

Different surfaces i.e. inlet, outlet, boundary of the solid are then named as per the required inlet and outlet for cold and hot fluids. The outermost wall is named as constant temperature surface. Generate the mesh and then close the window. Refresh and then update the project on the workbench. Then, open the setup. ANSYS Fluent Launcher will then open in a small window. Set the dimension as 3-D, option: Double Precision and click OK. The Fluent window will gradually open.

3.3.Solution:

3.3.1. Problem Setup

The mesh is automatically checked and the quality is obtained, if it is compatible. The type analysis is then changed - Pressure Based type. The velocity is changed to absolute and the time to steady state.

3.3.2. Models

Energy is set - ON position. As we prefer to analyze turbulent flow, so the viscous model is selected: “k-ε” model (2 equations).

3.3.3. Materials

The edit option is then clicked to add the water-liquid (for the hot and cold fluid) and copper (tubes) to the list - fluid and solid resp. from the specified fluent database.

Table 2. Input and Output Variables:

(i) Thermodynamic Properties

Constant	Notation	Units
Outlet temperature	T_{out}	K
Outlet pressure	P_{out}	Pa
Outlet density	ρ	kg/m^3
Mass flow rate	M	kg/s

(ii) Fluid Properties

Property	Notation	Units
Specific heat	Cp	-
viscosity	μ	-

(iii) Geometric Inputs

Component	Dimension	Notation	Unit
Helical Tube	Inner radius	R_I	mm
	Thickness radius	R_{th}	mm
	Outer radius	R_{out}	mm

	No. of Coils	n	-
	Pitch of helix	P	mm
	Coil dia	D	mm

(iv) Input fluid Conditions

Component	Constant	Notation	Unit
Heat exchanger as double helical tube	Hot fluid Inlet velocity	$v_{in, hot}$	m/s
	Cold fluid Inlet velocity	$v_{in, cold}$	m/s
	Cold fluid Inlet temperature	$T_{in, cold}$	Kelvin
	Hot fluid Inlet temperature	$T_{in, hot}$	Kelvin

3.3.4. **Cell zone conditions**

The parts are then assigned as water-liquid and copper as per the fluid/solid parts.

3.3.5. Boundary Conditions

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux. The outer wall constant temperature is taken as 313 K.

Table 3. Boundary Conditions

Boundary Condition Type	Velocity Magnitude		Turbulent Kinetic Energy	Turbulent Dissipation Rate	Temperature
Inner fluid Inlet	Velocity Inlet	1 m/s	0.01 m^2/s^2	0.1 m^2/s^3	343 K
Inner fluid Outlet	Pressure Outlet	-	-	-	-
Outer fluid Inlet	Velocity Inlet	2.5 m/s	0.01 m^2/s^2	0.1 m^2/s^3	293 K
Outer fluid Outlet	Pressure Outlet	-	-	-	-

3.3.6 Reference Values

The inner fluid inlet is selected from the list of “compute from”. The values are:

- Density = 998.2 kg/m^3
- Length 473.145 mm
- Temperature = 343 K
- Velocity = 1 m/s
- Viscosity = 0.001003 kg/m-s
- Ratio of specific heats = 1.4

3.3.7. Solution Methods

The solution methods are specified as follows:

- Scheme = Simplec
- Gradient = Least Square Cell Based
- Pressure = Linear
- Momentum = Power Law
- Turbulent Kinetic Energy = Power Law
- Turbulent Dissipation Rate = Power Law
- Energy = Second Order Upwind

3.3.8. Solution Control and Initialization

Under the relaxation factors the parameters defined are

- Pressure = 0.3 Pascal
- Density = 1 kg/m^3
- Body forces = 1 kg/m^2 s^2
- Momentum = 0.7 kg-m/s
- Turbulent kinetic energy = 0.8 m^2/s^2

The solution initialization method, is then set to Standard Initialization where the reference frame is set to the Relative cell zone. The inner fluid inlet is selected from the list and solution is initialized.

3.3.9. **Measure of Convergence**

For a stable convergence through the simulation, the criteria is made strict to get an accurate result. For this, the residuals are edited as per the given table:

Table 4 Residuals Variable

Table 4 Residuals Variable	Residual
x-velocity	10-4
y-velocity	10-4
z-velocity	10-4
Continuity	10-4
Specific dissipation energy/ dissipation energy	10-5
Turbulent kinetic energy	10-5
Energy	10-4

3.3.10. Run Calculation

The number of iterations is set to 750 and the solution is thus calculated and different contours, vectors and plots are thus obtained.

CHAPTER- 4

RESULTS AND ANALYSIS

4.1. D/d = 10 (d = 10 mm)

For Turbulent Flow,

Re = 2300 [1+ 8.6 * (d/D)$^{0.45}$]

= 9318.2 (for d/D = 1/10)

Now, Putting, Re = density * v * d / viscosity

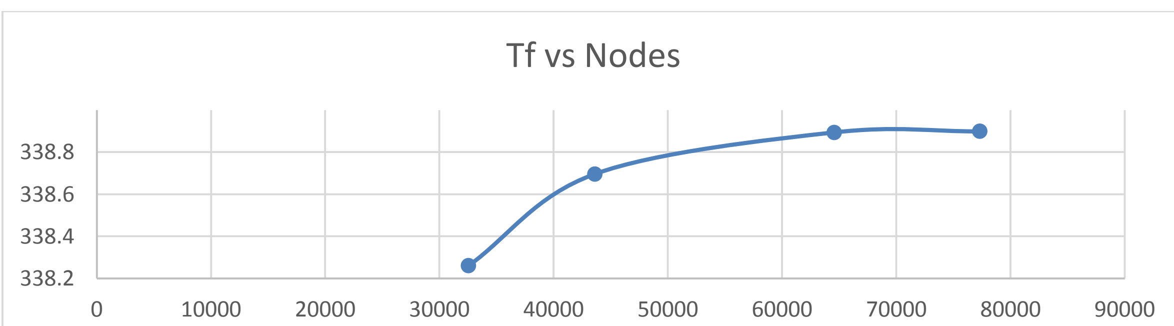
- ⇨ 9318.2 = 998.2 * v * 10^{-2} / 0.001003
- ⇨ => v = 0.936 m/s

So, for Turbulent flow, v > 0.936 m/s. So, the velocity of cold fluid is taken 2.5 m/s and velocity of hot fluid is taken from 1 m/s, increased in steps of 0.2 m/s to 2 m/s.

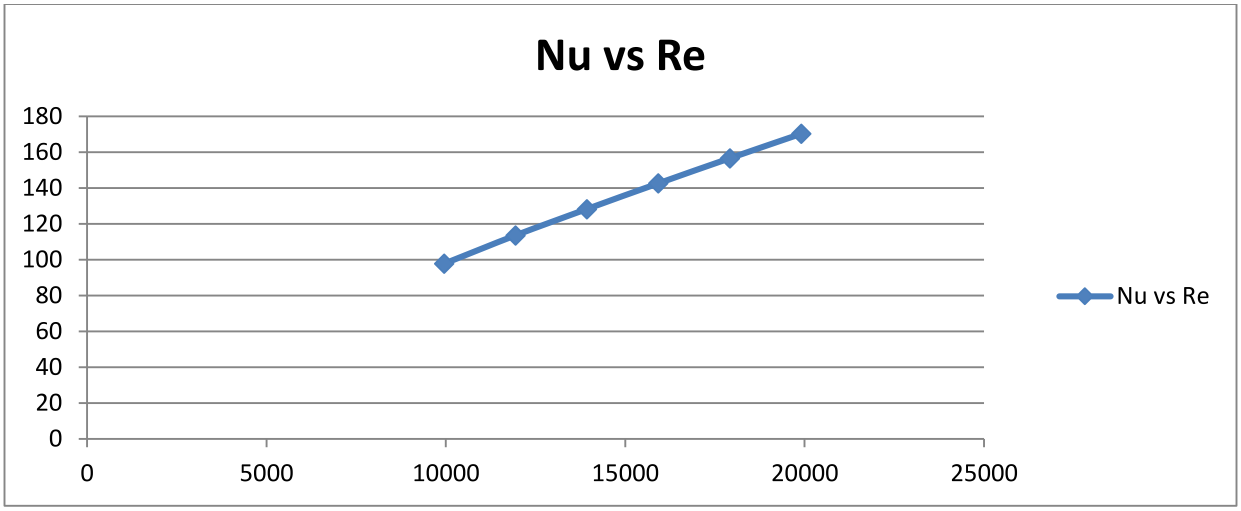
L_{eff} = n [(3.14*D)^2 + p^2]$^{0.5}$ = 473.145 m, (where n = 1.5, p = 30 mm)

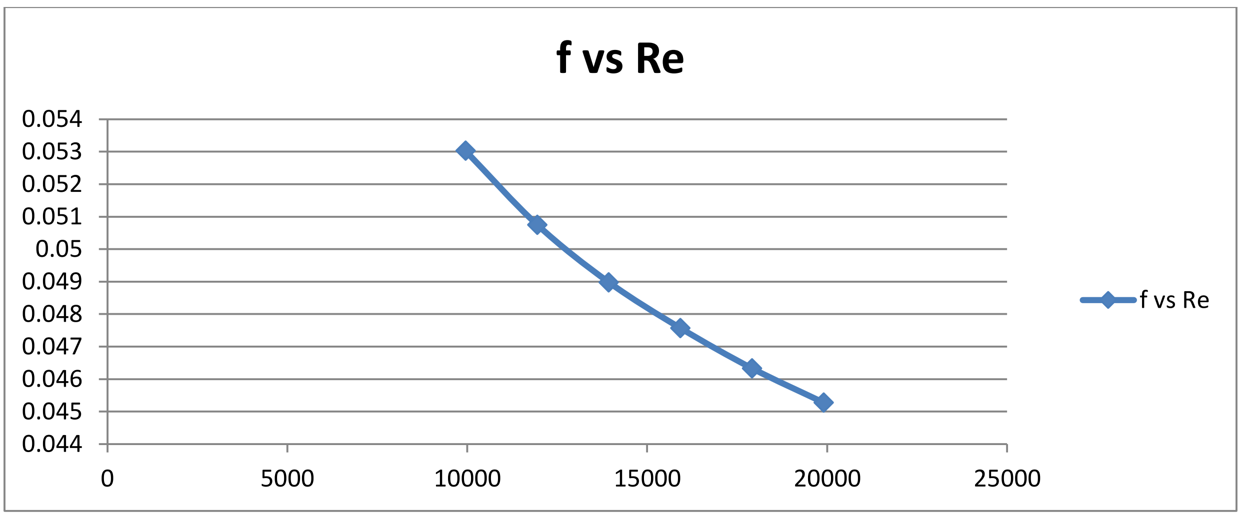
Grid Independent Test: (v = 1 m/s : hot fluid, cold fluid : v= 2.5 m/s)

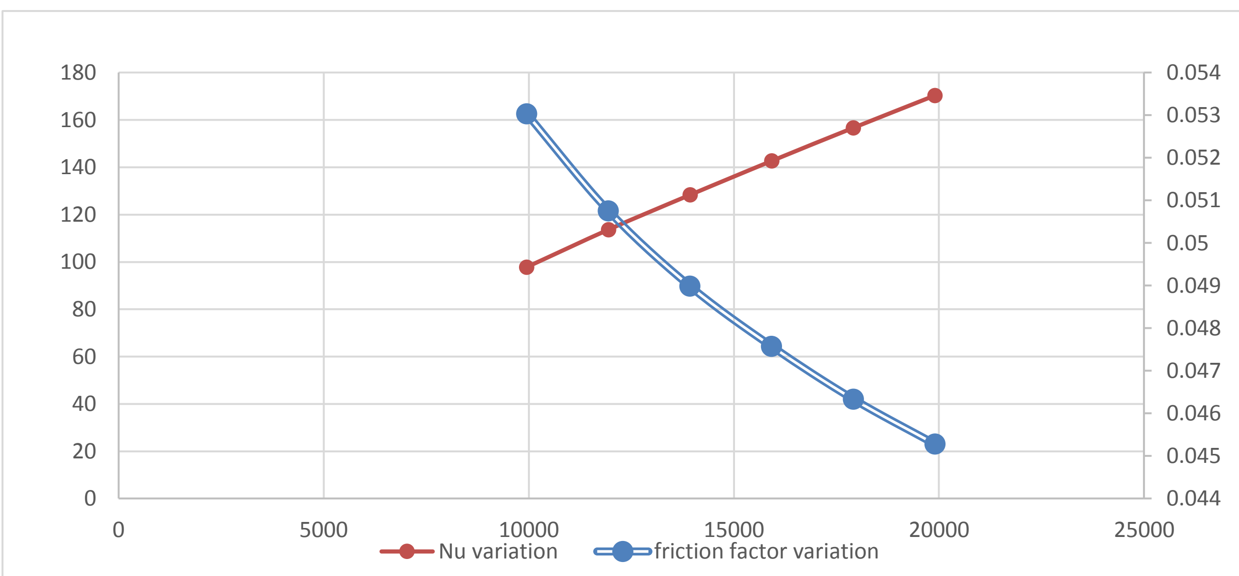
	No. of divisions	Nodes	Elements	Outlet Fluid Temp.
1	20, 24, 44, 48	32554	29904	338.26
2	24, 26, 50, 54	43598	106098	338.696
3	30, 32, 60, 62	64555	338407	338.894
4	40, 44, 80, 84	77314	357041	338.899

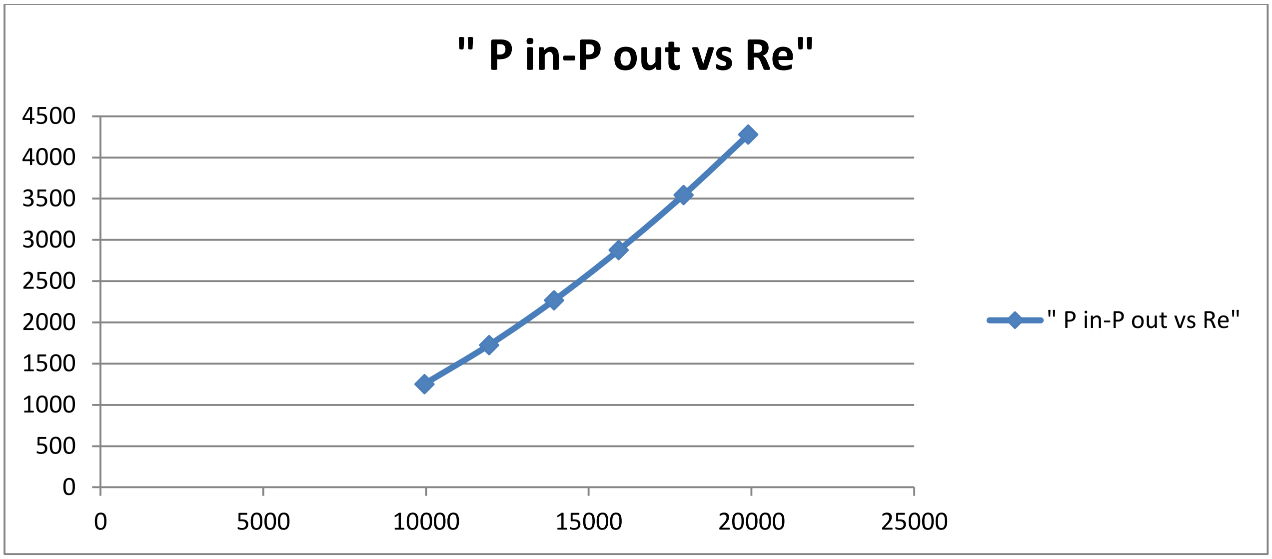


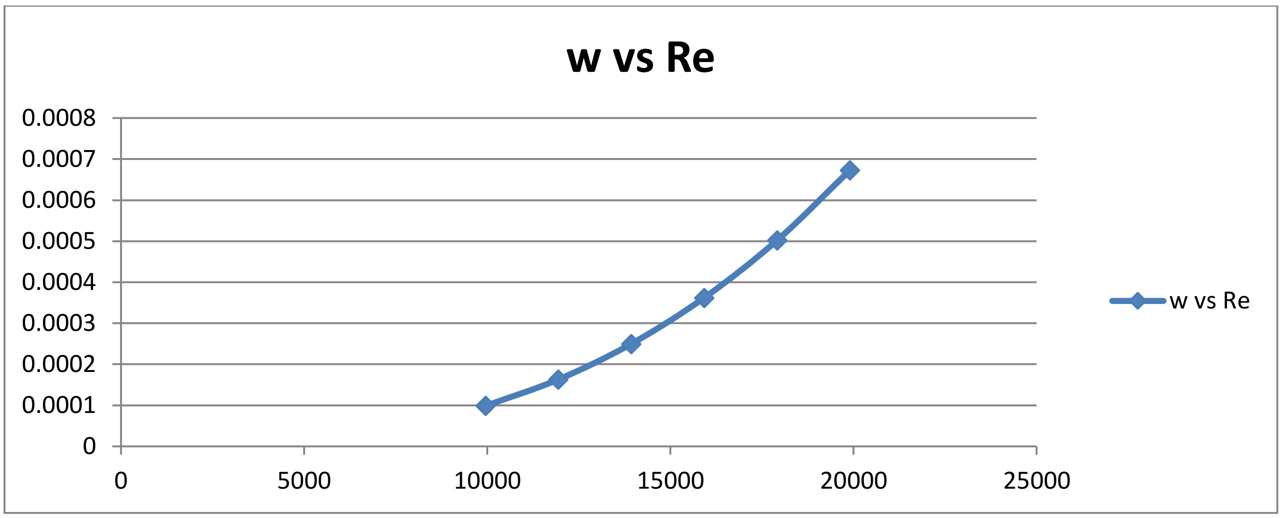
For Divisions (30, 32, 60, 62), it is Grid Independent, The graphs corresponding to it:











The Optimum heat transfer rate and minimum power loss or dissipation of mechanical energy at D/d =10 occurs corresponding to Reynolds no. 12250

4.2. D/d = 15 (d = 10 mm)

For Turbulent Flow,

Re = 2300 [1+ 8.6 * (d/D)$^{0.45}$]

= 8147.7 (for d/D = 1/15)

Now, Putting, Re = density * v * d / viscosity

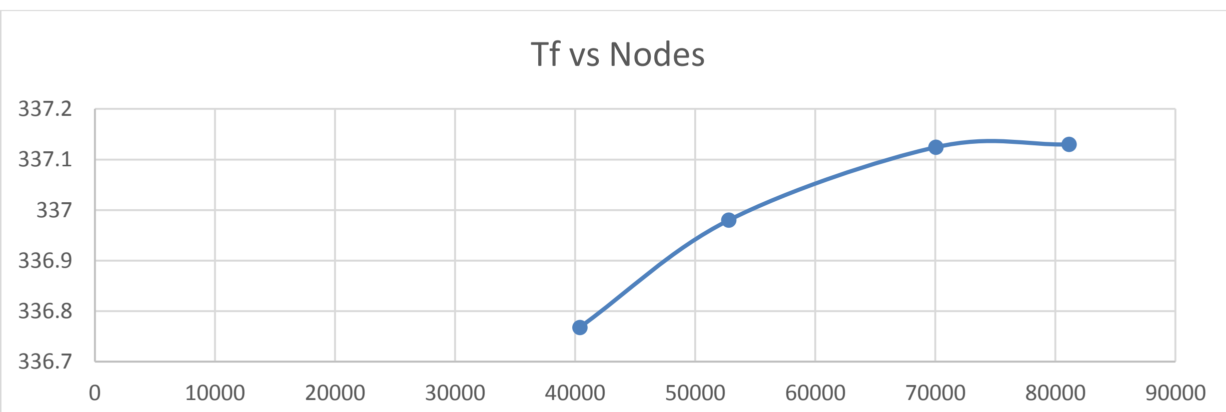
- ⇨ 8147.7 = 998.2 * v * 10^{-2} / 0.001003
- ⇨ v = 0.817 m/s

So, for Turbulent flow, v > 0.817 m/s. So, the velocity of cold fluid is taken 2.5 m/s and velocity of hot fluid is taken from 1 m/s, increased in steps of 0.2 m/s to 2 m/s.

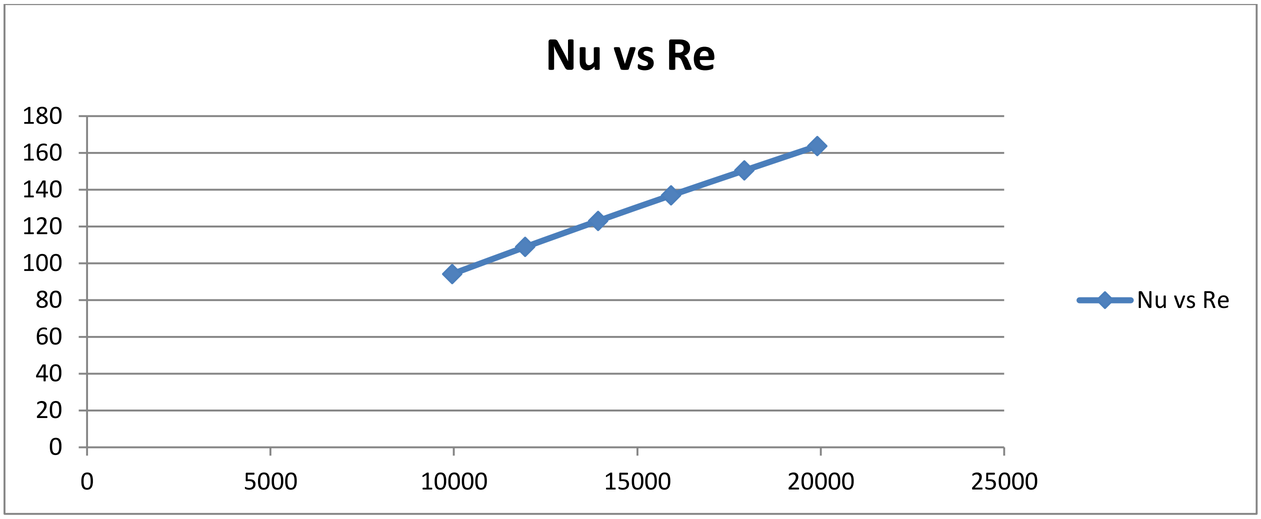
L_{eff} = n [(3.14*D)^2 + p^2]$^{0.5}$ = 708.38 m, (where n = 1.5, p = 30 mm)

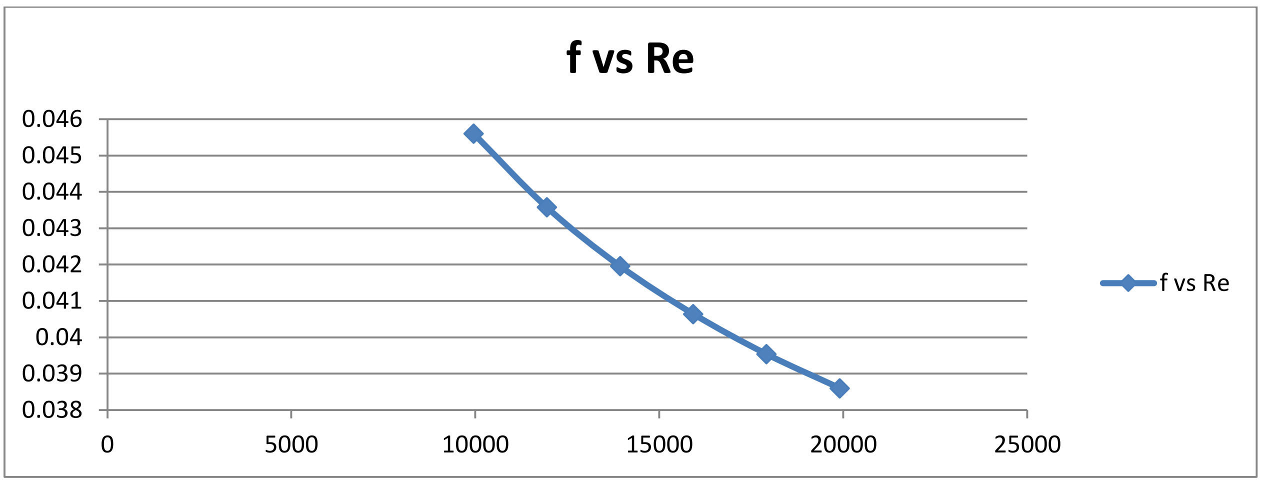
Grid Independent Test: (v = 1 m/s : hot fluid, cold fluid : v= 2.5 m/s)

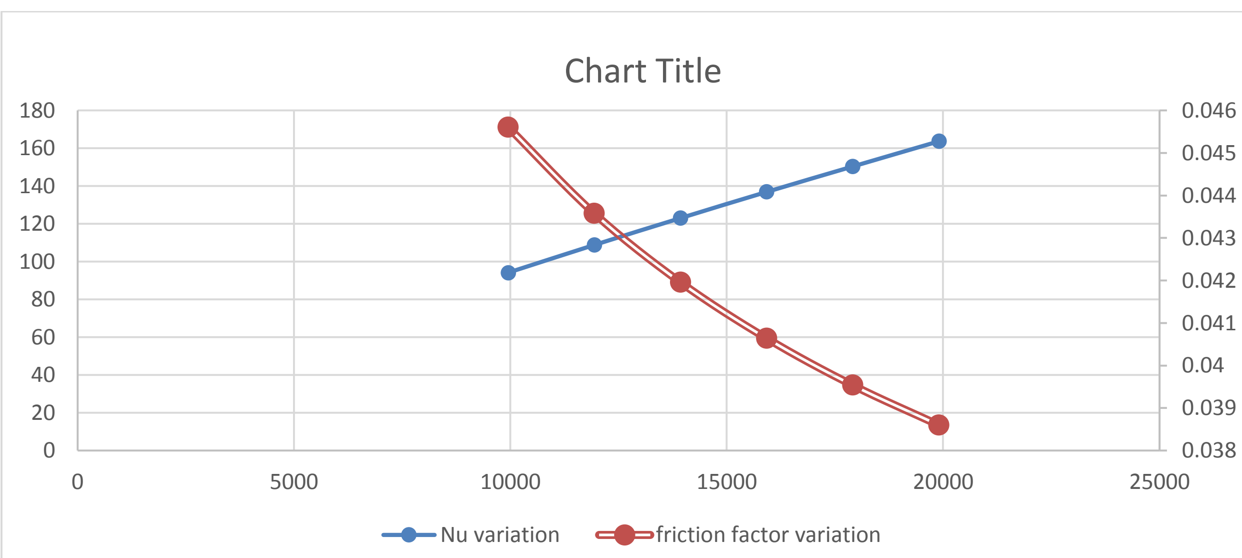
	No. of divisions	Nodes	Elements	Outlet Fluid Temp.
1	20, 24, 44, 48	40392	30415	336.768
2	24, 26, 50, 54	52798	116098	336.98
3	30, 32, 60, 62	70052	390200	337.1246
4	40, 44, 80, 84	81128	395345	337.13

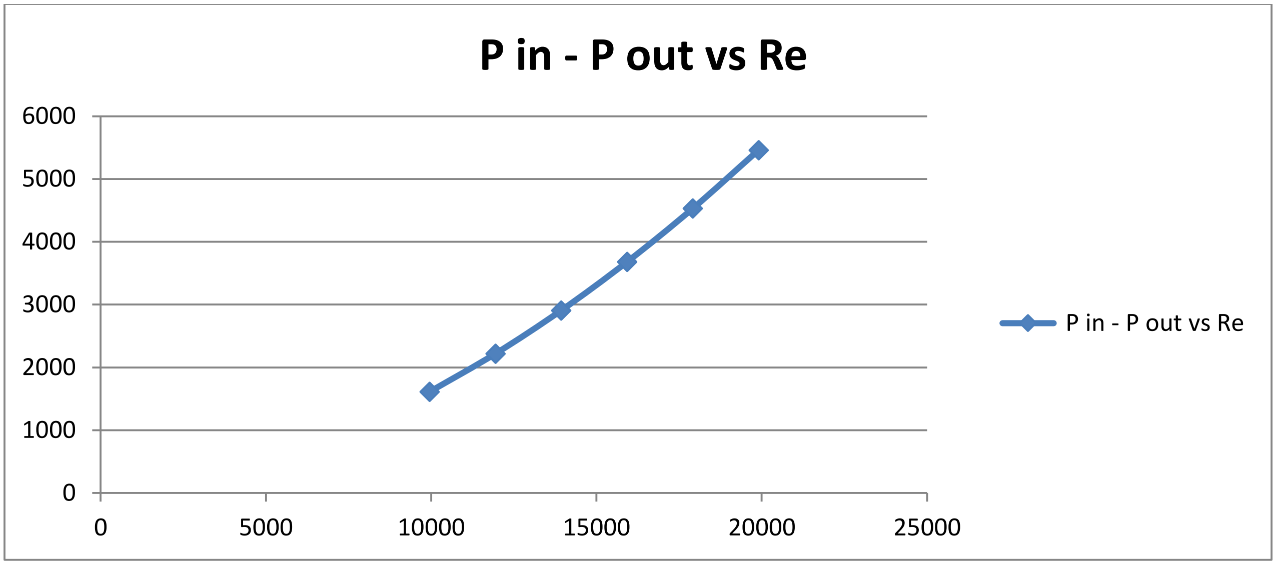


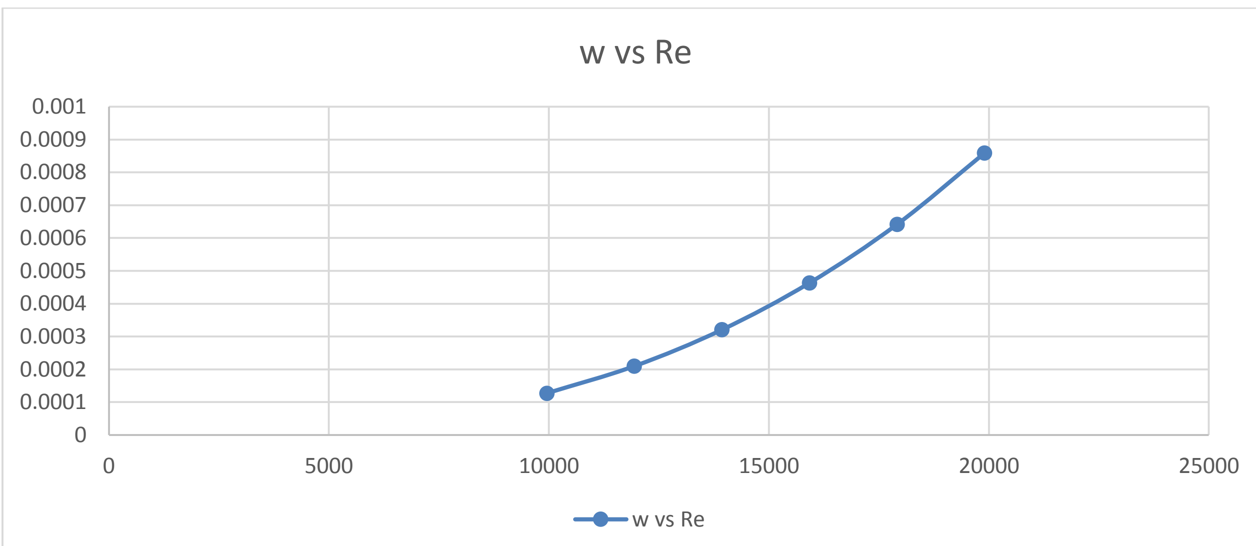
For Divisions (30, 32, 60, 62), it is Grid Independent, The graphs corresponding to it:











The Optimum heat transfer rate and minimum power loss or dissipation of mechanical energy at D/d =15 occurs corresponding to Reynolds no. 12750

4.3. D/d = 20 (d = 10 mm)

For Turbulent Flow,

Re = 2300 [1+ 8.6 * (d/D)$^{0.45}$]

= 7437.63 (for d/D = 1/20)

Now, Putting, Re = density * v * d / viscosity

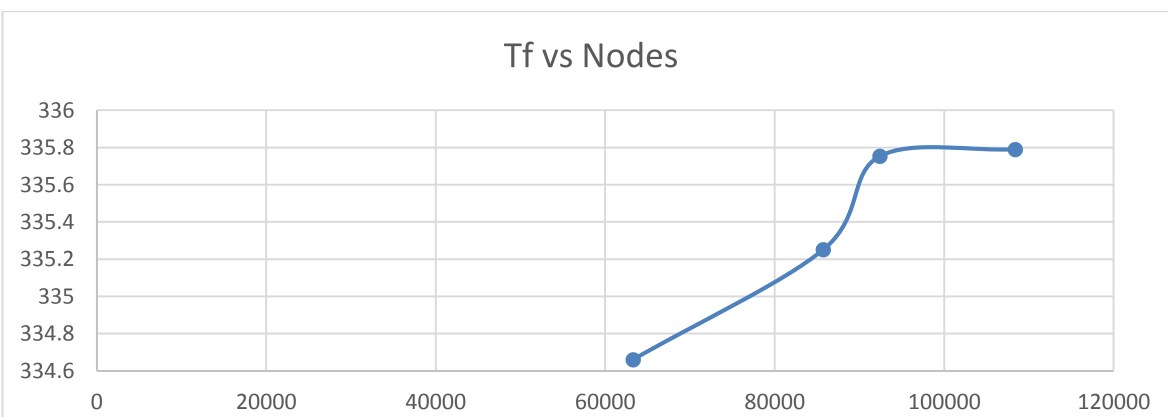
- ⇨ $7437.63 = 998.2 * v * 10^{-2} / 0.001003$
- ⇨ $v = 0.74734$ m/s

So, for Turbulent flow, v > 0.74734 m/s. So, the velocity of cold fluid is taken 2.5 m/s and velocity of hot fluid is taken from 1 m/s, increased in steps of 0.2 m/s to 2 m/s.

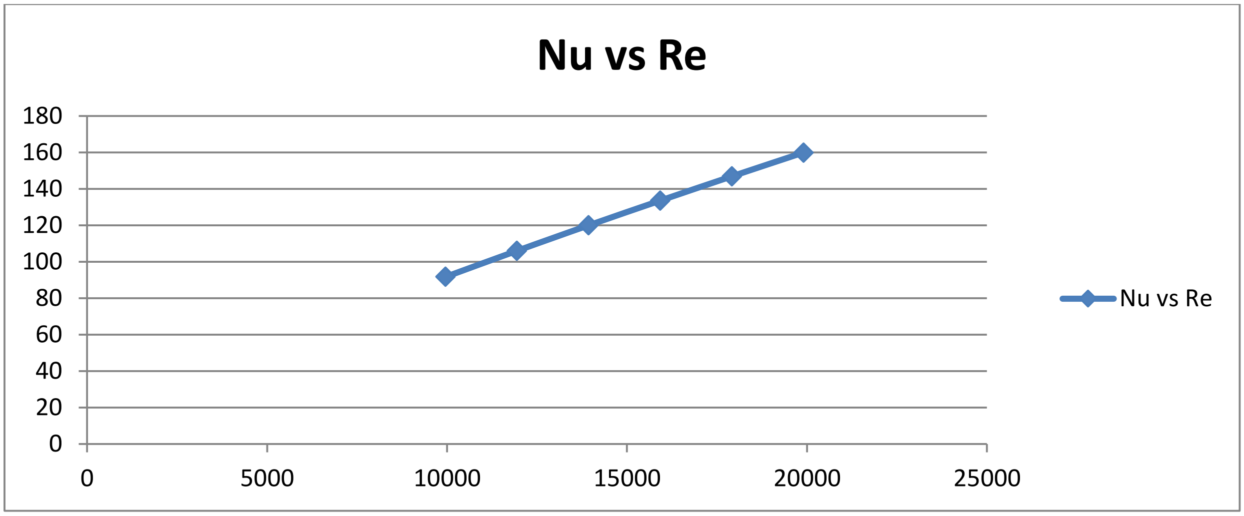
$L_{eff} = n [(3.14*D)^2 + p^2]^{0.5}$ = 943.074 m, (where n = 1.5, p = 30 mm)

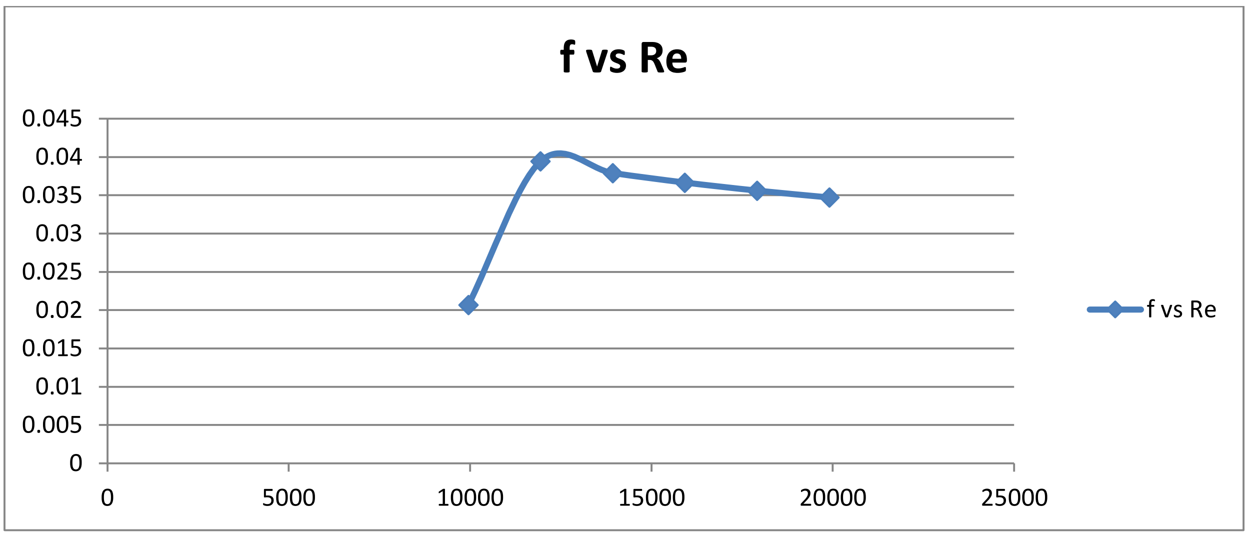
Grid Independent Test: (v = 1 m/s : hot fluid, cold fluid : v= 2.5 m/s)

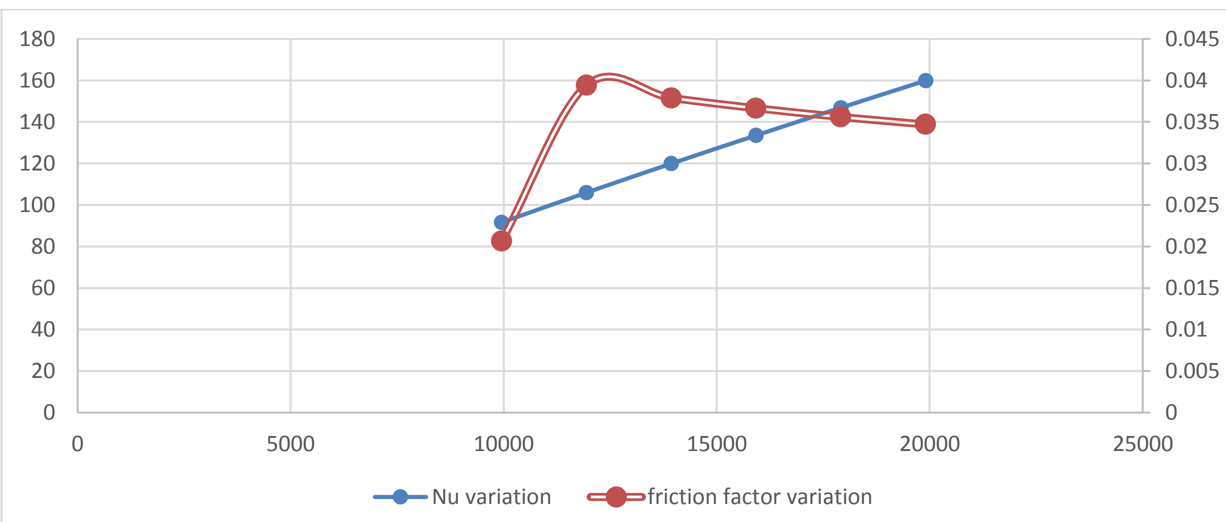
	No. of divisions	Nodes	Elements	Outlet Fluid Temp.
1	20, 22, 44, 48	63289	88512	334.66
2	24, 26, 50, 54	85738	179275	335.251
3	30, 32, 60, 62	92423	516630	335.753
4	40, 44, 80, 84	108405	578459	335.79

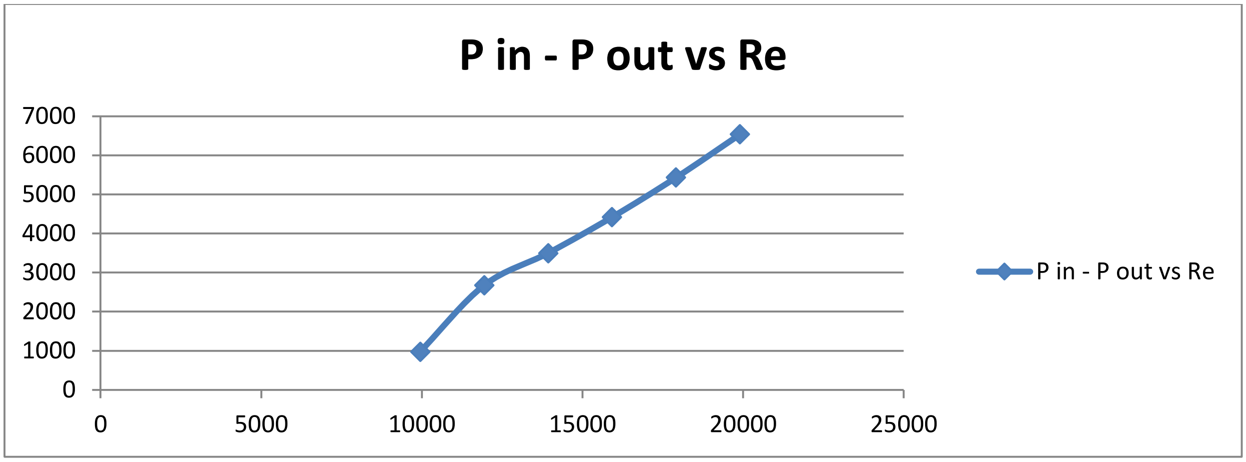


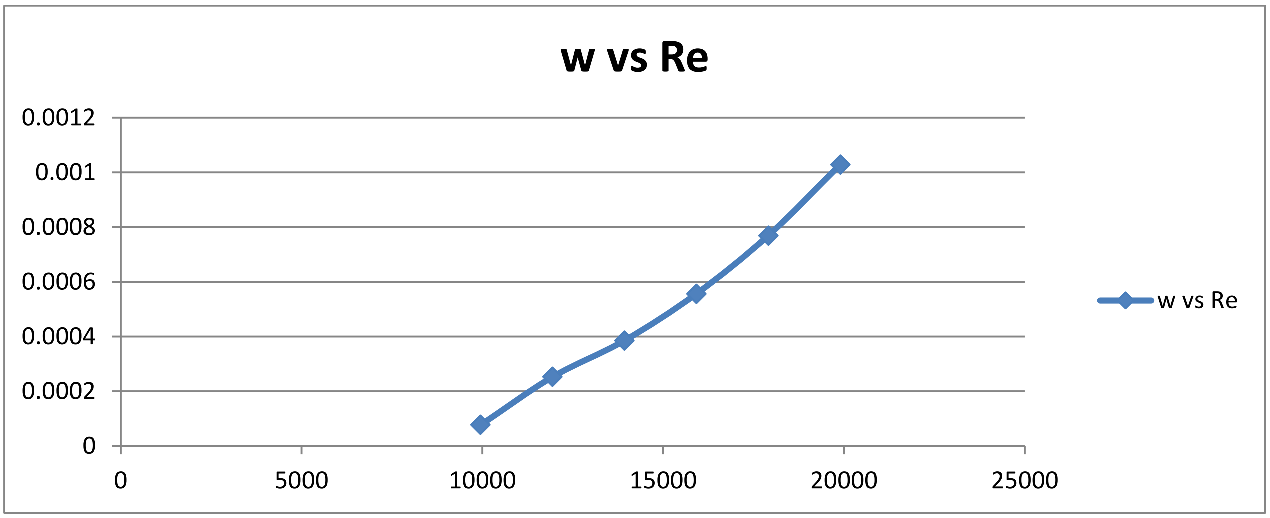
For Divisions (30, 32, 60, 62), it is Grid Independent, The graphs corresponding to it:











The Optimum heat transfer rate and minimum power loss or dissipation of mechanical energy at D/d =20 occurs corresponding to Reynolds no. 10000

4.4. D/d = 25 (d = 10 mm)

For Turbulent Flow,

Re = 2300 [1+ 8.6 * (d/D)$^{0.45}$]

= 6946.79 (for d/D = 1/25)

Now, Putting, Re = density * v * d / viscosity

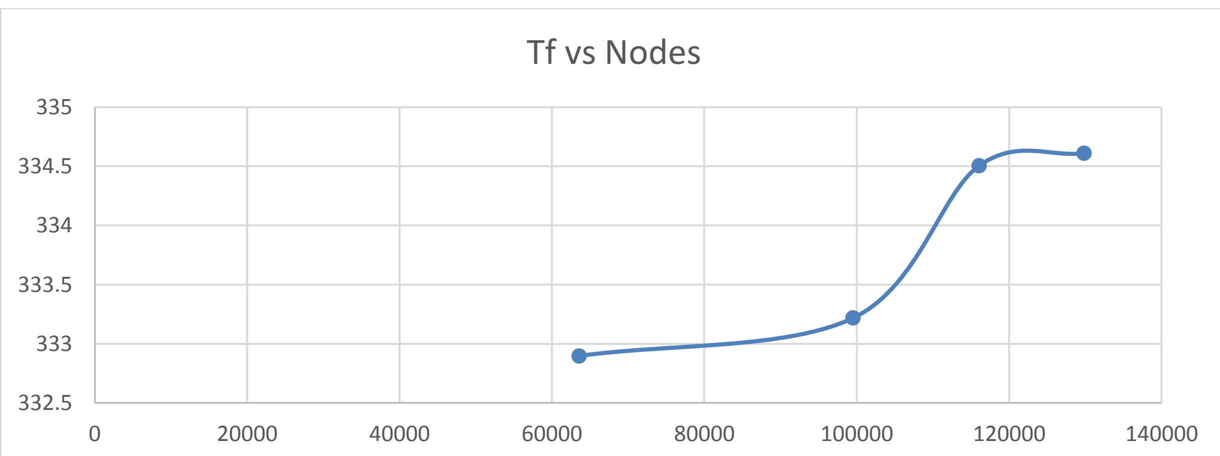
- ⇨ $6946.79 = 998.2 * v * 10^{-2} / 0.001003$
- ⇨ $v = 0.698$ m/s

So, for Turbulent flow, v > 0.698 m/s. So, the velocity of cold fluid is taken 2.5 m/s and velocity of hot fluid is taken from 1 m/s, increased in steps of 0.2 m/s to 2 m/s.

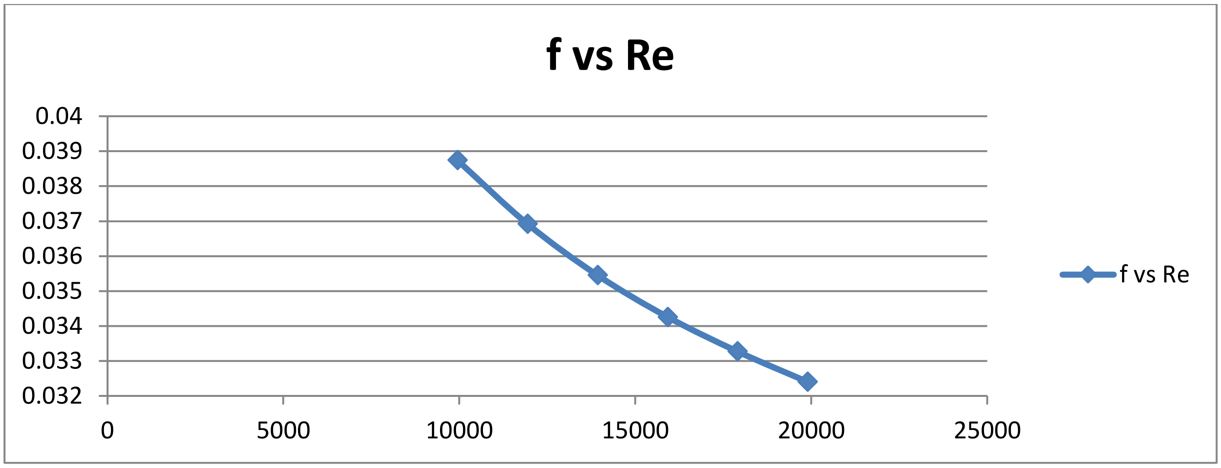
$L_{eff} = n [(3.14*D)^2 + p^2]^{0.5}$ = 1178.36 m, (where n = 1.5, p = 30 mm)

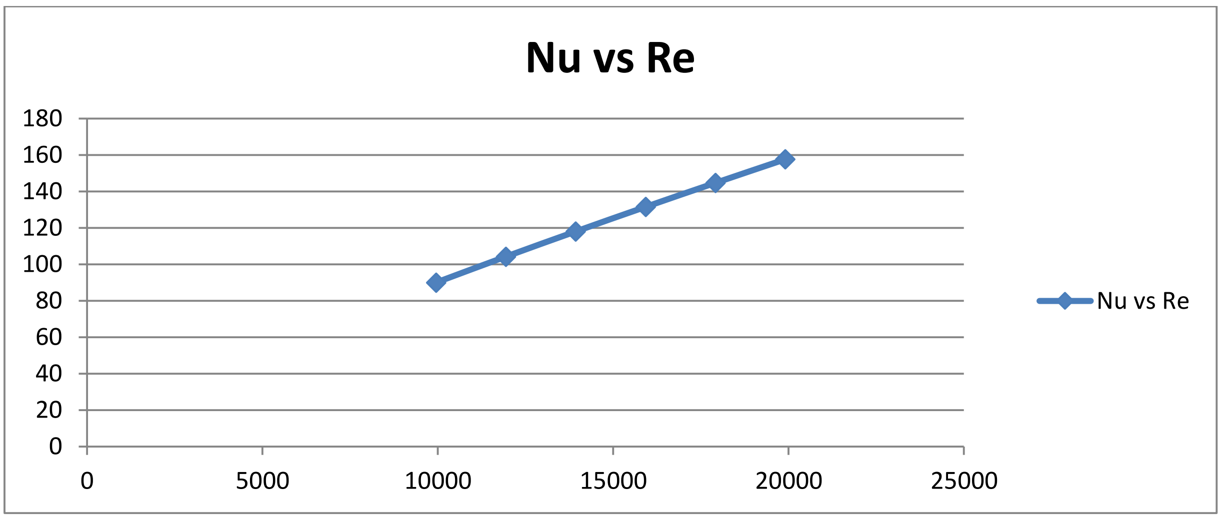
Grid Independent Test: (v = 1 m/s : hot fluid, cold fluid : v= 2.5 m/s)

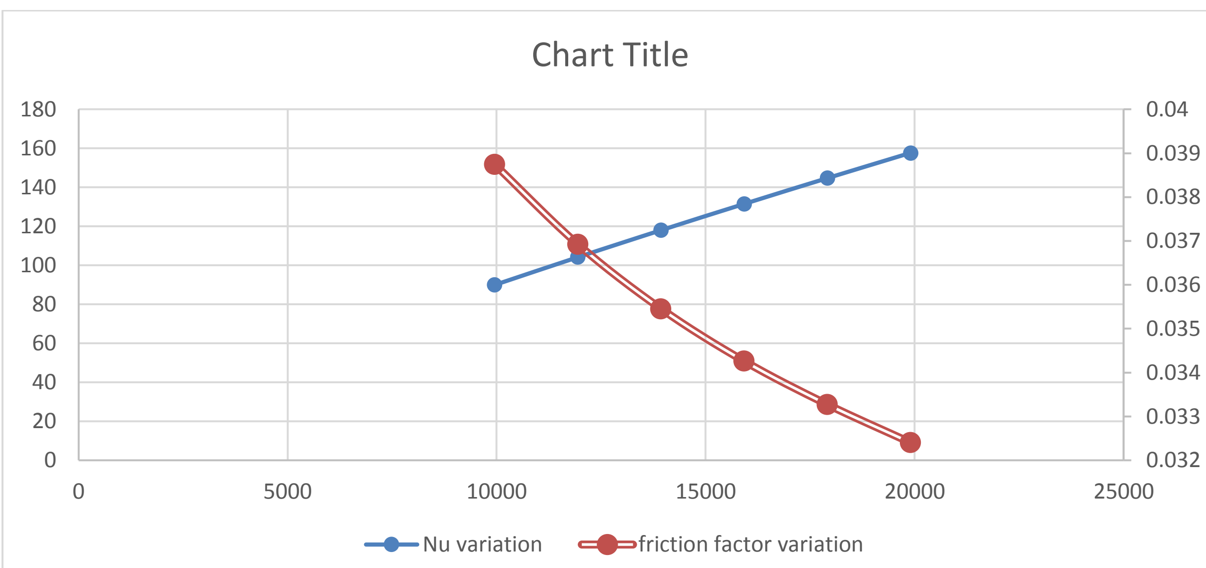
	No. of divisions	Nodes	Elements	Outlet Fluid Temp.
1	20, 22, 44, 48	63579	90599	332.897
2	24, 26, 50, 54	99512	192873	333.217
3	32, 34, 68, 70	116043	649525	334.5029
4	40, 44, 80, 84	129849	694785	334.61

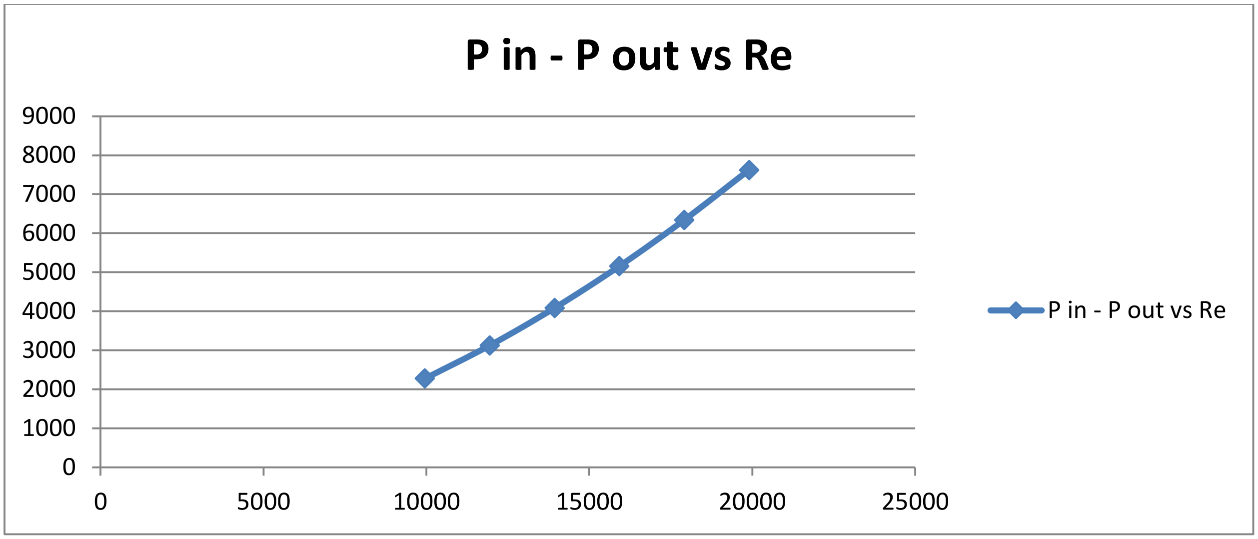


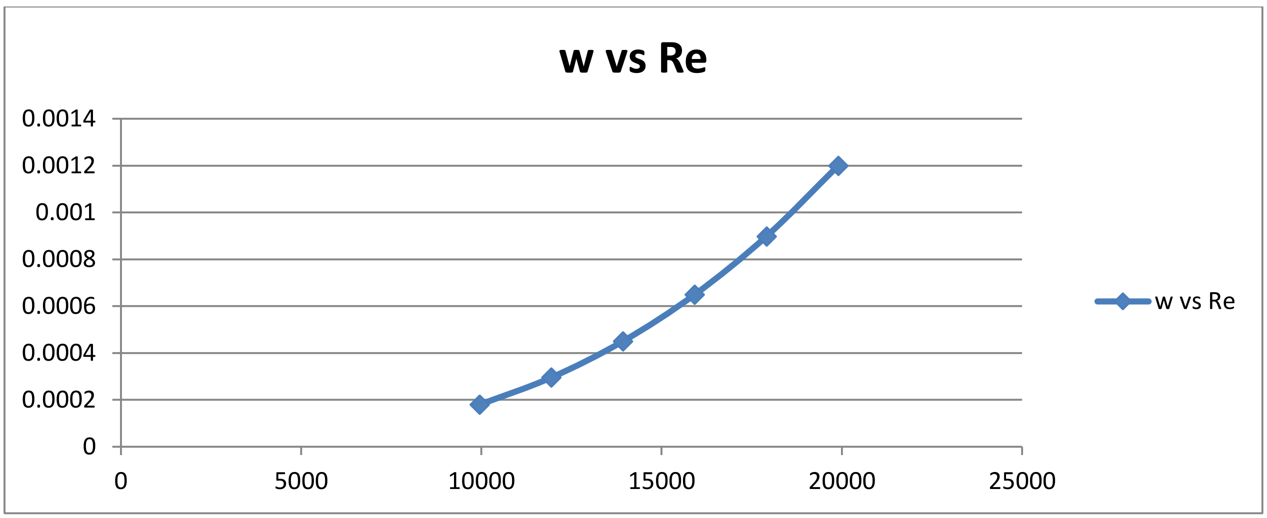
For Divisions (32, 34, 68, 70), it is Grid Independent, The graphs corresponding to it:











The Optimum heat transfer rate and minimum power loss or dissipation of mechanical energy at D/d =25 occurs corresponding to Reynolds no. 12500

4.5. D/d = 30 (d = 10 mm)

For Turbulent Flow,

Re = 2300 [1+ 8.6 * (d/D)$^{0.45}$]

= 6580.768 (for d/D = 1/30)

Now, Putting, Re = density * v * d / viscosity

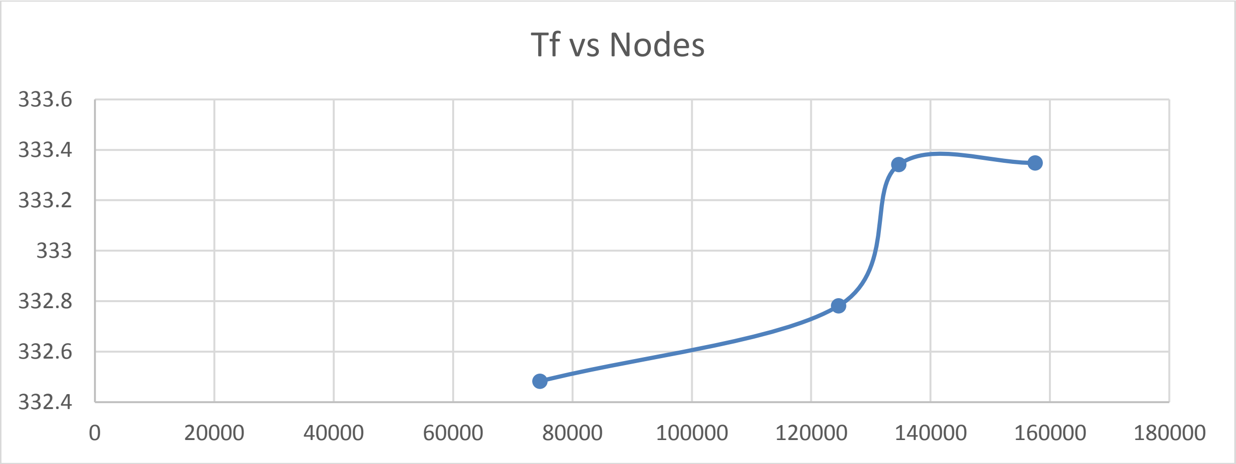
- ⇨ 6580.768 = 998.2 * v * 10^{-2} / 0.001003
- ⇨ v = 0.6612 m/s

So, for Turbulent flow, v > 0.6612 m/s. So, the velocity of cold fluid is taken 2.5 m/s and velocity of hot fluid is taken from 1 m/s, increased in steps of 0.2 m/s to 2 m/s.

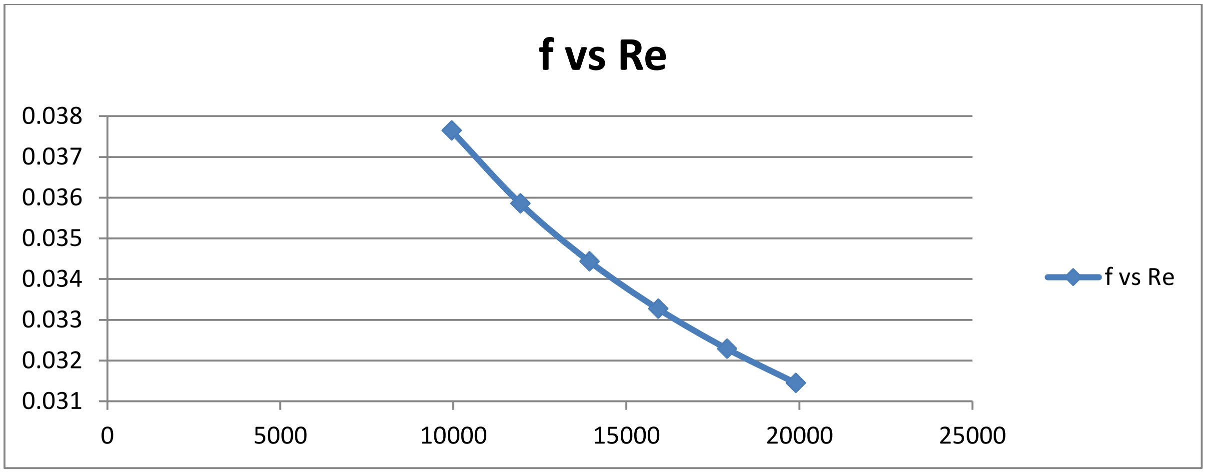
L_{eff} = n [(3.14*D)^2 + p^2]$^{0.5}$ = 1414.437 m, (where n = 1.5, p = 30 mm)

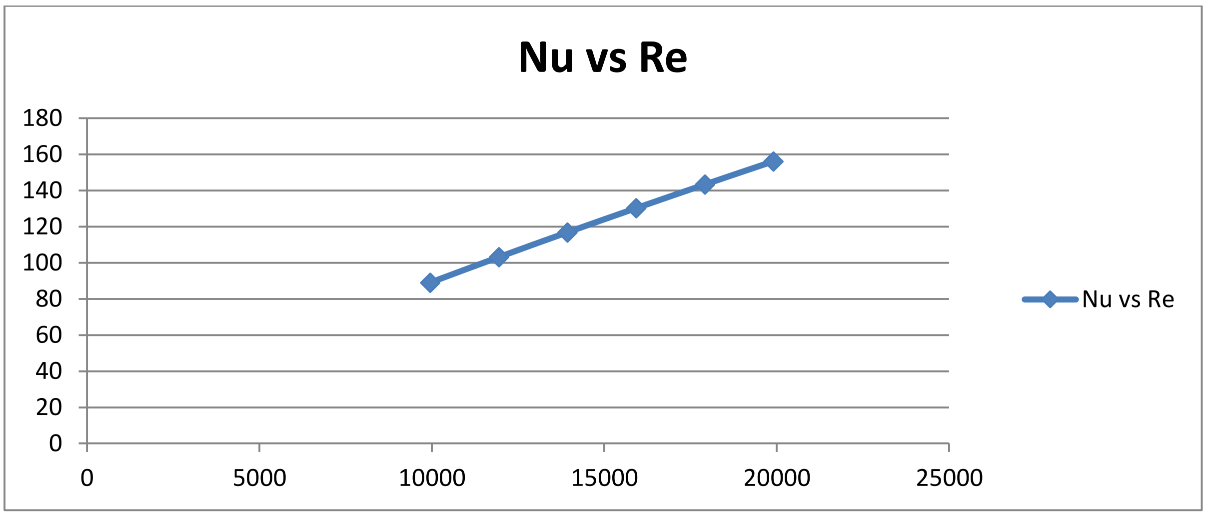
Grid Independent Test: (v = 1 m/s : hot fluid, cold fluid : v= 2.5 m/s)

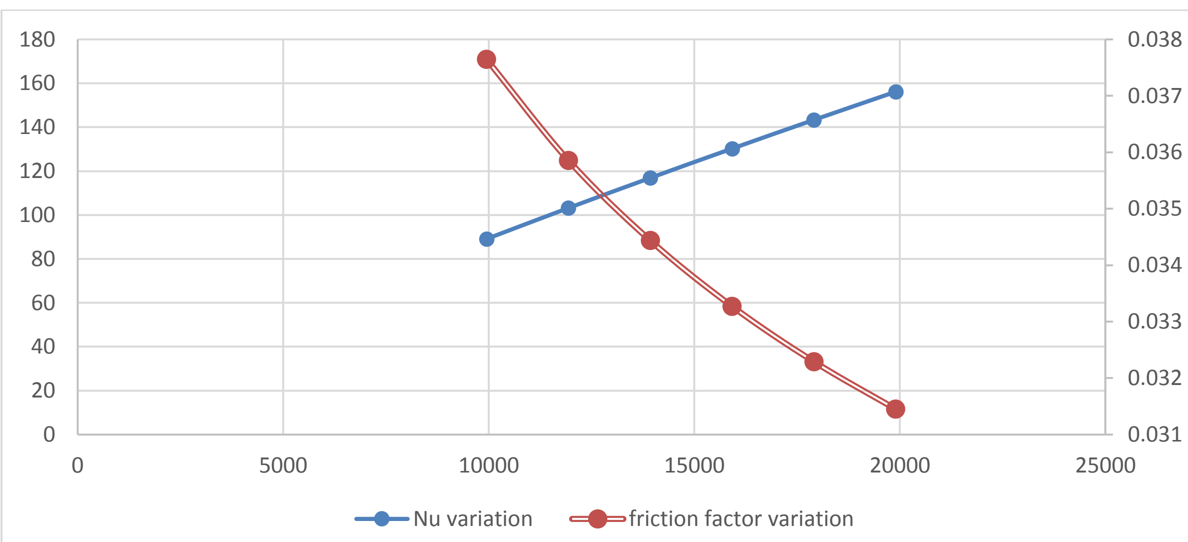
	No. of divisions	Nodes	Elements	Outlet Fluid Temp.
1	20, 22, 44, 48	74563	102920	332.482
2	24, 26, 50, 54	124575	116870	332.781
3	32, 34, 68, 70	134678	717085	333.341
4	40, 44, 80, 84	157484	984747	333.348

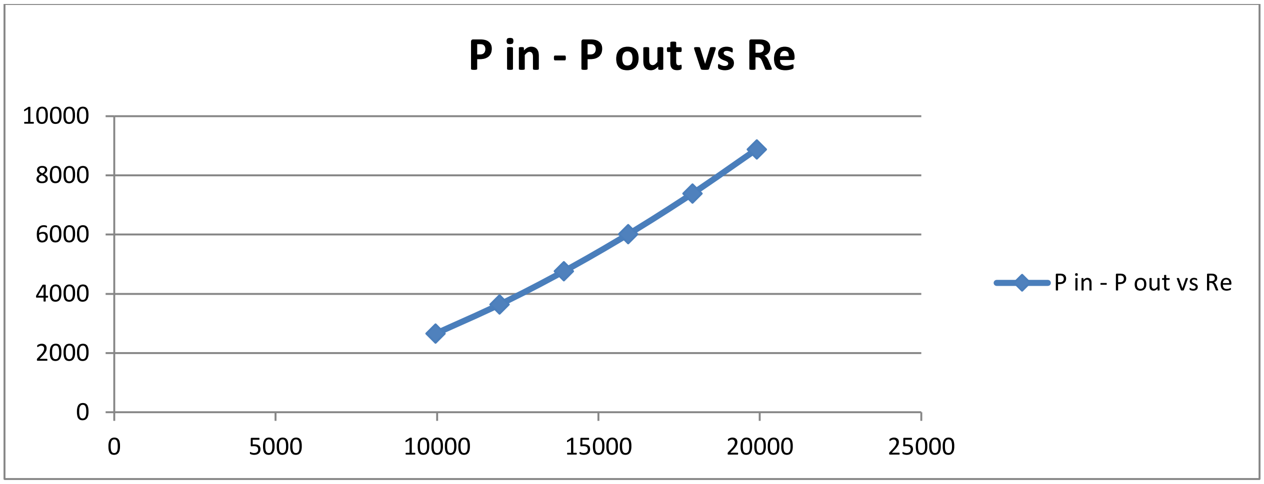


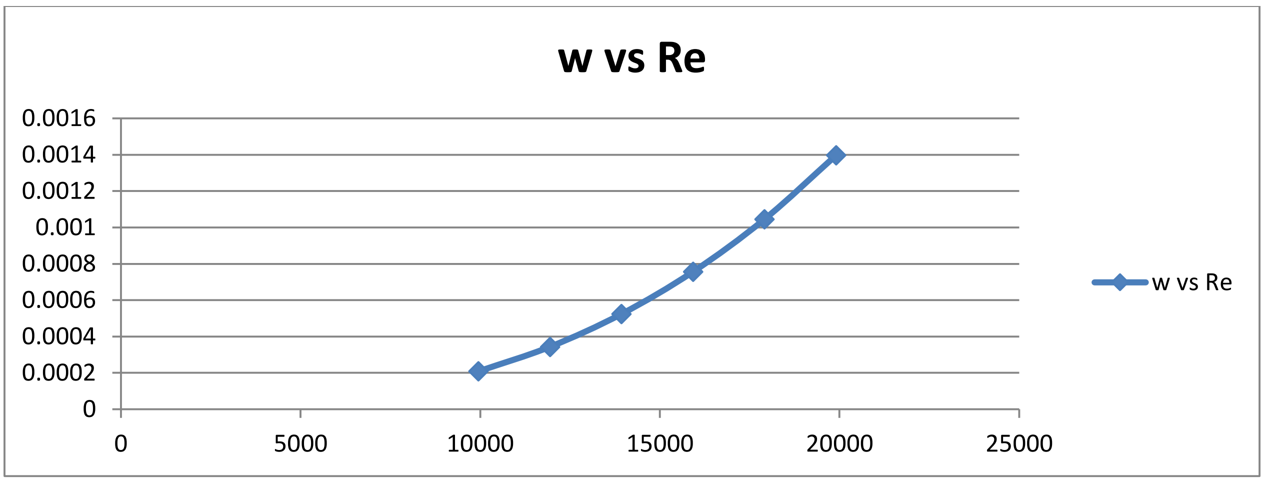
For Divisions (32, 34, 68, 70), it is Grid Independent, The graphs corresponding to it:











The Optimum heat transfer rate and minimum power loss or dissipation of mechanical energy at D/d =30 occurs corresponding to Reynolds no. 13000

CHAPTER- 5

5. CONCLUSION

Thermodynamic Optimization in heat transfer of a concentric coiled tube-in-tube heat exchanger under constant wall temperature condition, based on Fluid–Fluid heat transfer is discussed. CFD analysis was carried out and their variation on thermal and hydraulic characteristics were analyzed, with varying Reynolds number (hot fluid) and varying tube-to-coil diameter ratios for a given flow velocity of cold fluid. The analysis was carried with Ansys 13.0 Fluent, for turbulent counter-flow with fluid water. The correlations for heat transfer and drop in pressure were analyzed. Thus, Nusselt number and friction factor were also calculated. Graphs were plotted between Nusselt number, friction factor, pressure drop and power loss with Reynolds number. The point where the friction fraction intersects with the Nusselt number is the point where the heat transfer is optimum, corresponding to that Reynolds number. Beyond that Reynolds number, the friction factor decreases rapidly, hence the pressure drop increases and so the power loss also increases. Various velocity and temperature contours were also obtained. Hence, we found the optimum value of Reynolds number for the corresponding tube-to-coil diameter ratios. It thus minimizes the degradation of thermal energy and viscous dissipation of mechanical energy.

6. REFERENCS

1. Experimental and CFD estimation of heat transfer in helically coiled heat enchangers – S.D. Sancheti, Dr. P.R. Suresh, 2012.

2. Development of heat transfer coefficient for concentric helical coil heat exchanger – Rahul Kharat, Ntin Bhardwaj, R.S. Jha. International Journal of Thermal Sciences 48, 2009 (2300-2308).

3. Thermal hydraulic characteristics of air-water two-phase flows in helical pipes. - J.S. Jayakumar , S.M Mahajani, J.C Mandal, Kannan N. Iyer and P.K. Vijayan , 2010 (501-512).

4. Experimental study of mixed convection of heat exchanger in vertically concentric helical coil heat exchanger - N. Ghorbani, H. Teherain, M. Gorhi and H. Mirgolbabaei, 2010 , Experimental thermal and Fluid science 34 (900-905).

5. Numerical studies of a double pipe helical heat exchanger - Timothy J. Rennie, Vijaya G.S. Raghavan, Applied thermal engineering 26, 2006 (1266-1273).

6. Boundary vorticity method for laminar forced convection heat transfer in curved pipes - Akiyama, M. and Cheng, K.C., 14(10): (1659–1675).

7. Thermodynamic optimization of a coiled tube heat exchanger under constant wall heat flux condition – Ashok k. Satapathy - Science direct (1122-1126)

8. Tata McGraw-Hill Education, 1978 – Heat and mass transfer – P.K. Nag, 3rd edition.

9. Tata McGraw-Hill Education, 1989 – Heat transfer – Yunus A. Cengel, 3rd edition.